Experimental and CFD Analysis of a Perforated Inner Pipe Muffler for the Prediction of Backpressure

Sudarshan Dilip Pangavhane [#1]; Amol Bhimrao Ubale [#2]; Vikram A Tandon [*1]; Dilip R Pangavhane [*2]
[#] Symbiosis International University, Pune, Maharashtra, India
[#1] sdpangavhane@outlook.com
[#2] amol_ubale@yahoo.co.in
[*1]Automotive Research Association of India, Pune, Maharashtra, India
[*1] tandon.nvh@araiindia.com
[*2]Prestige Institution of Engineering and Science, Indore, Madhya Pradesh, India
[*2] drpangavhane@yahoo.co.in

***Abstract* - Backpressure is essential for the performance of a silencer. Pressure drop of exhaust system includes losses due to piping, silencer, and termination. The most critical component regarding backpressure of any commercial muffler is cross flow perforated tube in which the diameter of the perforated tube hole and porosity of the perforations are most critical. In this paper the effect of change in dimensions of perforation diameter and change in porosity of internal tube is investigated using CFD analysis and the simulated data is compared with experimental results. It is found that the porosity of the muffler has pronounced effect on the Backpressure. The Backpressure reduced almost by 75% if the porosity is doubled. Also, if the diameter of the hole increases the backpressure decreases sharply by 40%. The change in diameter of holes has remarkable effect on back pressure. CFD and Experimental values are in good agreement with each other.**

Key words: - Back pressure, perforation, CFD analysis, mufflers.

I. INTRODUCTION

Engine exhaust noise is controlled through the use of silencers and mufflers. Well-designed exhaust systems collect exhaust gases from engine cylinders and discharge them as quickly and silently as possible. Primary system design considerations include:

1. Minimizing resistance to gas flow (back pressure) and keeping it within the limits specified for the particular engine model and rating, to provide maximum efficiency.
2. Reducing exhaust noise emission to meet local regulations and application requirements.
3. Providing adequate clearance between exhaust system components and engine components, machine structures, engine bays, enclosures and building structures to reduce the impact of high exhaust temperatures on such items.
4. Ensuring the system which does not overstress engine components such as turbochargers and manifolds with excess weight. As overstressing can shorten the life of engine components.
5. Ensuring the exhaust system components which are able to reject heat energy as intended by the original design.

The parametric and shape optimization techniques presented by Key Fonseca de Lima and others produces good designed and optimized reactive mufflers [1]. Reactive silencers rarefy the sound mainly by increasing and decreasing the cross section of the fluid flow path when the fluid is travelling from one chamber to another. Every change in cross-section results in a reflection of part of the wave with a 180°- phase shift. This difference in phase results in cancellation of part of the incoming wave [2].

Silencer's performance is mainly dependent on the values of backpressure. Pressure drop of exhaust system includes losses due to piping, silencer, and termination. High backpressure can cause a decrease in engine efficiency or increase in fuel consumption, overheating, and may result in a complete shutdown of the engine potentially causing significant damage.

Backpressure has a significant effect on performance of two stroke engine. The pressure in two-stroke engine can be considered atmospheric when the cylinder filling (scavenging) is superimposed to the exhaust phase, i.e., with the exhaust port opened to the external atmosphere. The effectiveness of the cylinder filling with fresh charge depends on small differences of pressure, and the two stroke engine can only tolerate small amounts of exhaust back pressure.

In two stroke engines, which use the crankcase as a scavenging pump, the dynamic effect plays a fundamental role in filling the cylinder with fresh charge. Every time the exhaust flow meets a section increase, a negative pressure is generated and propagates in the opposite direction of flow with the speed of sound. On the other hand, if the flow finds a restricted section, a positive pressure wave always propagates in the opposite direction with the speed of sound. According to acoustic theory, the propagation of waves depends strictly on the length and section of ducts, volumes, and logically on the speed of sound. [3]

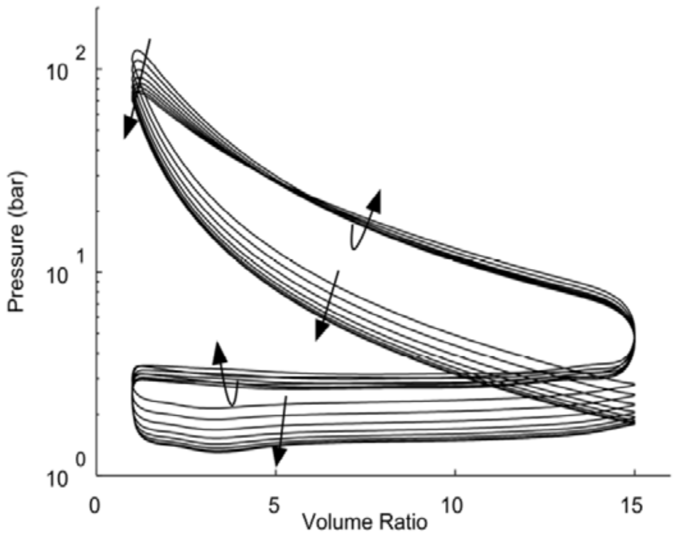

Fig. 1. The P-V diagram for a cylinder, for various Backpressure. Note the Logarithmic Scale on the Pressure Axis. Arrows show direction of change with increasing Backpressure [4]

In four stroke engines the most obvious effect is the increase in size of the pumping loop as the back pressure increases, due to the extra work done by the piston on the gas in pumping it out of the cylinder during the exhaust stroke as shown in figure 1. This represents the extra work that must be done by the engine as the back pressure increases, in addition to meeting the constant load demand. Although the maximum cycle pressure decreases due to the reduced compressor pressure ratio, the engine pressure ratio remains effectively constant. The gradient of the power stroke curve also decreases with increased back pressure. This is due to the increase in the burn duration that occurs with reduced maximum cylinder pressure [4].

Thus, excessive backpressure can adversely affect performance, resulting in reduced power and increased fuel consumption, exhaust temperatures and emissions. It will reduce exhaust valve and turbocharger life. Also it is imperative that exhaust backpressure is kept within specified limits for those engines subject to emissions legislation.

II. EXPANSION CHAMBER ELEMENTS HAVING INFLUENCE ON BACKPRESSURE REFER FIGURE 2)

A conventional muffler of internal combustion engine is mostly constructed as a mixture or combination of perforated ducts, baffle or perforated baffle, expansion chamber, etc., [5]

A. Head pipe Or The Inlet Pipe

In general, a longer head pipe will bring about more bottom-end power at the cost of peak power. A small head pipe generally brings stronger peak power and subtracts bottom-end. Practically tapered head pipes are relatively more difficult to manufacture and costly too, so they are rare in conventional vehicles. Tapered head pipes have proven to boost performance and relatively ease pipe tuning in their main area of influence.

B. Convergent and Divergent Cones

The length, volume and taper of the divergent cone strongly influence the amount of peak power. A relatively short, steeply tapered, divergent cone creates high peak power. What happens after an engine's power peaks is nearly as important as the peak itself. Controlling power after the peak i.e. the overrun is the divergent cone's job. A relatively longer, gently tapered final cone will give more overrun. A short, steep final cone gives less.

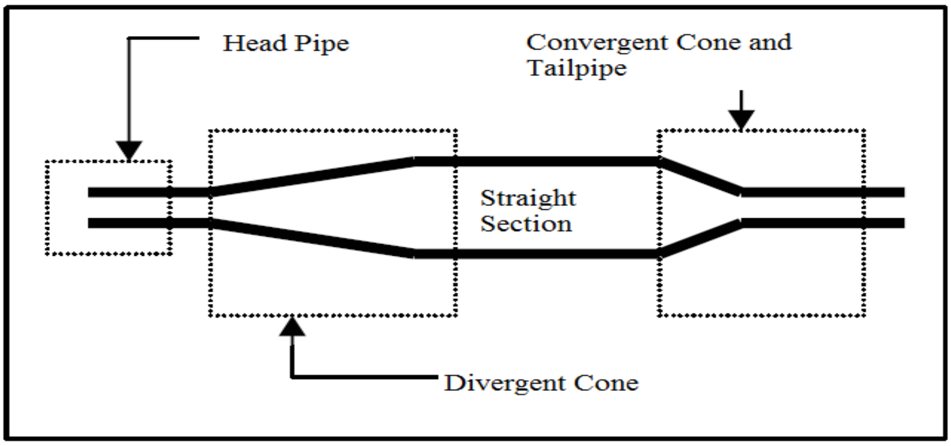

Fig. 2. Components of an Expansion Chamber

C. Straight Section

The length or volume adjustments are compensated for "ideal" head pipe, divergent cone, convergent cone and tailpiece/silencer dimensions at the pipes mid section. The pipe's straight section can be enlarged, shortened or lengthened to bring about the same results like "ideal" designs.

D. The Tailpiece Or Outlet Pipe

Tailpiece size and its length influences peak power and bottom-end, and can even affect an engine's resistance to pistons. In general, smaller tailpiece diameters create more peak horsepower but increase the occurrence of thermal breakdown because it accumulates the exhaust heat. Big tailpiece diameters increase bottom-end at the cost of peak power and excessively large tailpiece diameters can reduce performance at all engine speeds due to insufficient back pressure. Tailpiece length is also important, because it is the part of the total pipe length and volume. Generally, longer tailpiece is used when low and midrange power is required. The tailpipe itself acts as a resonant cavity that couples with the muffler cavity. The noise reduction capability of a muffler gets changed, if designed tail piece is not used. [3]

E. Perforations on Internal connecting tubes or cross flow perforated tube

The most critical component regarding backpressure of any commercial muffler is cross flow perforated tube. The diameter and porosity of the perforations are most critical [4]. A smaller hole diameter but higher porosity creates high peak power with acceptable backpressure value. While a larger hole diameters with higher porosity reduce performance due to insufficient back pressure. Sound energy gets dissipated considerably while moving through the perforations and it adds to the total attenuation in perforated tubes [6]. The perforated pipes are complex acoustic impedance and are evaluated using simple empirical relations [7].

So, in this paper the effect of change in dimensions of perforation diameter and change in porosity on internal tube is investigated using CFD analysis. Also, to validate the CFD results three commercially available mufflers are tested at Automotive Research Association of India (ARAI), Pune.

III. SOLVER, MODELS AND SCHEMES USED FOR CFD ANALYSIS

The physical model of this test setup would be passing air at fixed mass flow rate through the muffler and measuring pressure drop across the muffler. The pressure drop is same as the back pressure acting on the engine through the exhaust system. The time conditions implemented are steady state. The Mass flow input is given as a constant number ranging from 0.0398 Kg/s to 0.0693 Kg/s, so the flow is subsonic flow.

A flow becomes incompressible for low speed, that, is $M<0.3$ for air and compressible for Mach number $M \geq 0.3$, depending on pressure and density changes relative to the local speed of sound. So, the computational schemes are dictated by various physical conditions: Viscosity, incompressibility and compressibility of the flow.

The solver implemented was pressure based as it is used for incompressible flows to keep the pressure field from oscillating, which may arise due to difficulties in preserving incompressibility conditions or conservation of mass as the speed of sound become much higher than convection velocity components. Thus to preserve the incompressibility conditions, pressure correction algorithms have to be used, for pressure based solvers.

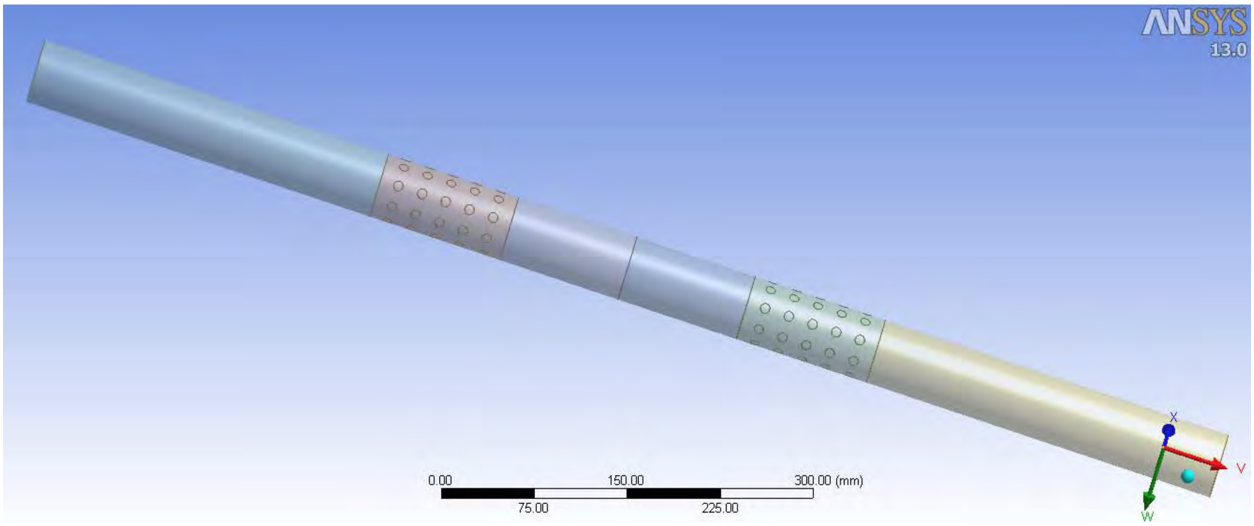

Fig. 3. Muffler with Perforated Internal Pipe with Hole Diameter 7.5 mm

Pressure velocity coupling refers to numerical algorithm which uses a combination of continuity and momentum equations to derive the equation for pressure when using the pressure based solver. From the available algorithms Semi Implicit with splitting of operators (SIMPLE) is used as it provides solution without iterations, with large time steps and less computing efforts.

In fluent solver variable are stored at the center of grid cells. To solve the transport equations we need to know the values of quantities at the faces of control volume and gradient of these quantities in the cell. The quantities are namely momentum, turbulent kinetic energy, turbulent dissipation rate and energy.

So, for interpolating the cell center data various interpolating schemes are available in fluent. This interpolation in ANSYS-FLUENT is called upwind schemes. Upwinding means that the face value is derived from quantities in the cell upstream, or “upwind,” relative to the direction of the normal velocity. Of the available schemes first order upwind method is used for interpolating the values of these quantities so as to reduce computation effort. Gradients are needed not only for constructing values of a scalar at the cell faces, but also for computing secondary diffusion terms and velocity derivatives. The gradient of a given variable is used to discretize the convection and diffusion terms in the flow conservation equations. Thus, to determine the gradient of these variables Green –Gauss node based method is implemented. This scheme reconstructs exact values of a linear function at a node from surrounding cell-centered values on arbitrary unstructured meshes by solving a constrained minimization problem, preserving a second-order spatial accuracy. The node-based gradient is more accurate than other gradient schemes especially on (skewed and distorted) unstructured meshes, at a higher computing power cost.

For calculating the cell-face pressure, as pressure based solver is implemented, standard interpolation scheme is used.

Most of the engineering flows are turbulent. Turbulence occurs when velocity gradients are high, resulting in disturbances in flow domain as a function of space and time. Turbulent flow arises in contact with walls or in between two neighboring layers of different velocities. With velocity gradient increasing, the flow becomes rotational, leading to vigorous stretching of vortex lines, which can only be supported in three dimensional. So, turbulent flows are always physically three dimensional. In turbulent flows large and small scales of continuous energy spectrum, which are proportional to the size of eddy motions, are mixed. Here, eddies are overlapping in spaces with large ones carrying the small ones. In this process the turbulent kinetic energy transfer from larger eddies to smaller ones, with the smallest eddies eventually dissipating into heat through molecular viscosity.

So, it is necessary to model the turbulence model appropriately. The Standard K-Epsilon model (SKE) is the used as it is accurately represent engineering turbulence for industrial applications. Also, it is Robust and reasonably accurate for a wide range of applications. Also it accurately represents low speed incompressible flows in isotropic turbulence.

Standard K-Epsilon model is a Two-equation turbulence models and allows the determination of both, a turbulent length and time scale by solving two separate transport equations. It is based on model transport equations for the turbulence kinetic energy (K) and its dissipation rate (ε). The model transport equation for K is derived from the exact equation, while the model transport equation for ε is obtained using physical reasoning. In the derivation of the K-ε model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard K-ε model is therefore valid only for fully turbulent flows [8], [9].

IV. EFFECT OF CHANGE IN POROSITY AND CHANGE IN DIAMETER OF PERFORATION HOLE ON BACKPRESSURE

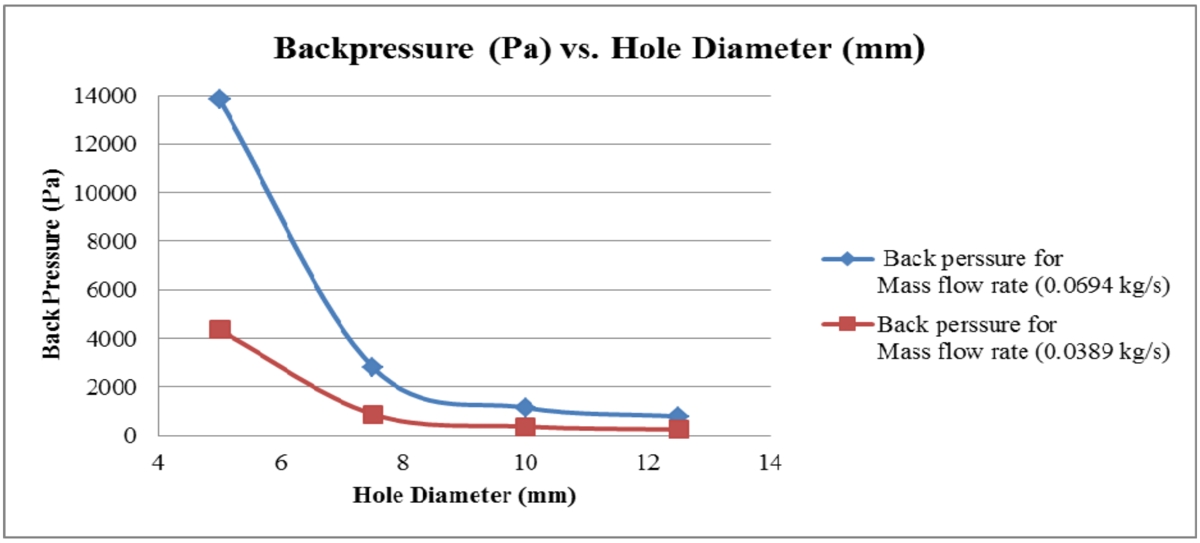

Fig. 4. Graph of Backpressure (Pa) vs. Perforated Hole Diameter (mm)

From fig-4 it is observed that for the smallest hole diameter of 5 mm the back Pressure is as high as 13,837 Pa. If we increase the diameter of the hole Back Pressure rapidly falls down and it is lowest i.e. 788 Pa for the hole diameter 12.5 mm. The pressure drop is very large which is 75% of highest backpressure for first two hole diameters viz. 5 mm and 7.5 mm. For other hole diameters the pressure drop is small but significant.

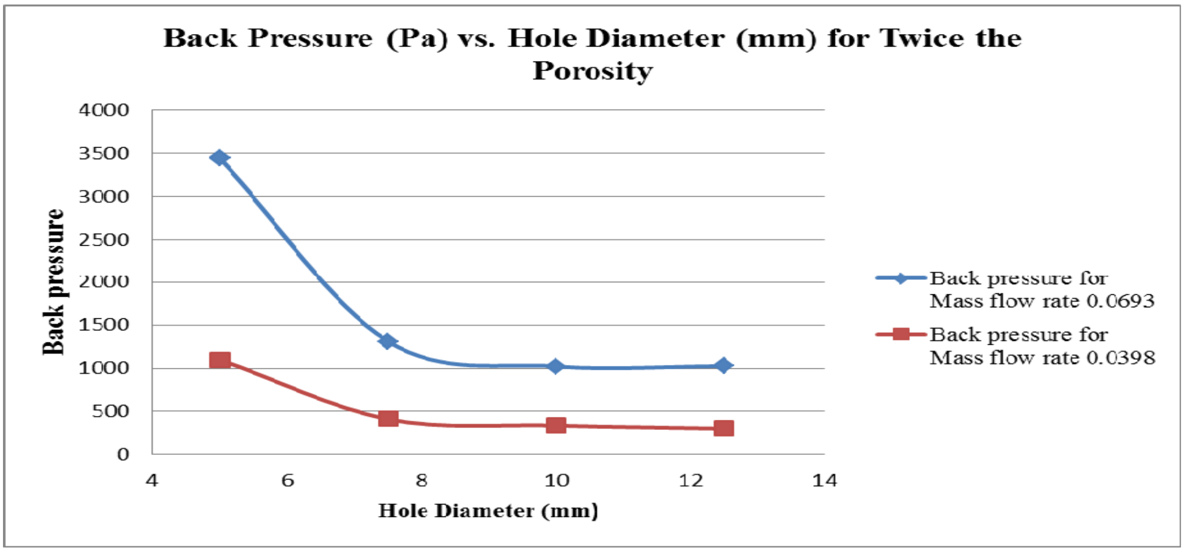

Fig. 5. Graph of Back Pressure vs. Hole Diameter (mm) for Double the Porosity

When the porosity is doubled than the conventional, backpressure drops by 75% for first two hole diameters. While for other hole diameters it is fairly the same value with a difference of 20 Pa to 75 Pa. Thus it can be seen that the backpressure value is high for small diameters as compare to bigger diameter holes even if the porosity is doubled. But for higher diameters the Backpressure value remains the same even when the porosity is doubled.

V. EXPERIMENTAL ANALYSIS AND CASE STUDY

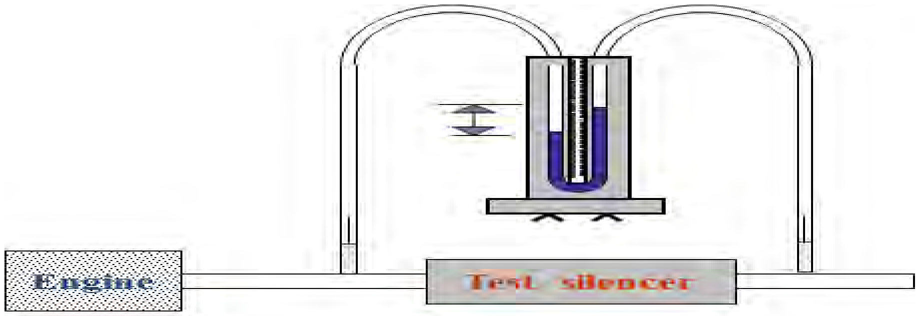

Fig. 6. Schematic Diagram of experimental setup for Backpressure Measurement with a U tube manometer

The schematic Diagram of experimental setup for Backpressure Measurement with a U tube manometer is shown in figure 6. The pressure taps are located at a location Upstream of the muffler and another one at downstream of the muffler. The location of the pressure tap should be at such a location where the length and cross section of the pipe should be continuous enough i.e. there are no bends of change in cross section of the pipe.

CFD analyses of the muffler tested in experiment have been performed for two mass flow rates. All the solver conditions, turbulence modeling and boundary conditions have been kept same as in the previous analysis of perforated inner pipe mufflers.

The CFD results are as shown in Figure 7. A part of the actual test setup is shown in figure 8. CFD and experimental values are in good agreement with each other. The difference in the CFD values and Experimental values is because various parameters like pressure drop due to friction are not considered. Also CFD is an approximate method. So, in order to achieve higher accuracy it is necessary to make the mesh more dense which will result in unnecessary computation time. The Pressure and velocity contours for all the three muffler test are shown in figure 9 to figure 17.

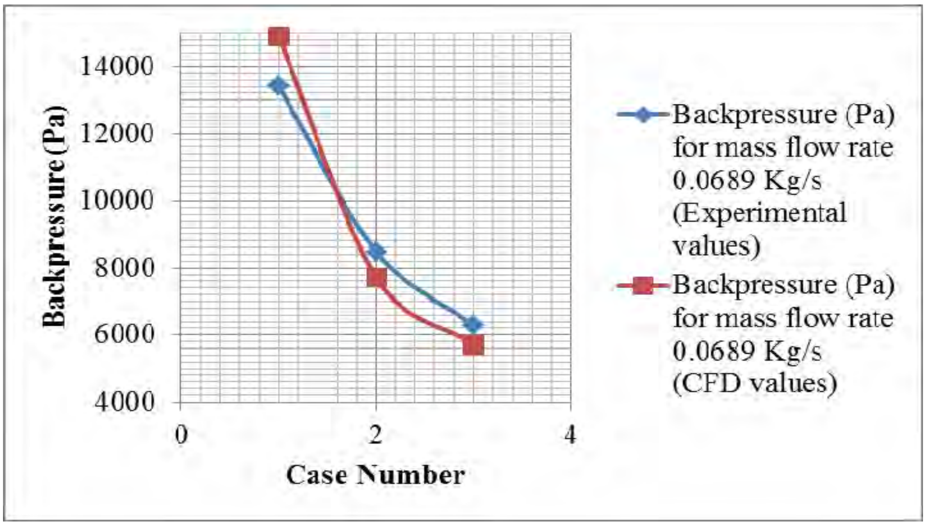

Fig. 7. Graph of Backpressure (Pa) for comparison of Experimental and CFD values

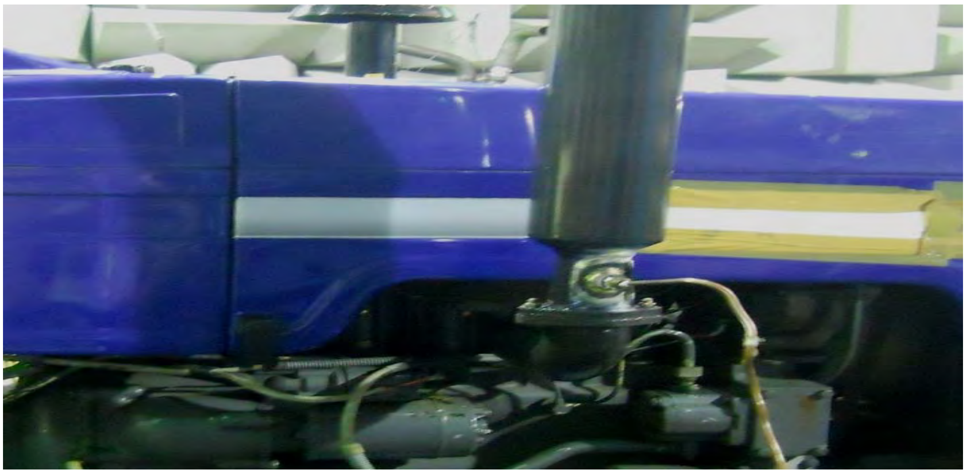

Fig. 8. Location of Pressure Tap on the Muffler Tested in Lab

Fig. 9. Muffler CAD Model 1 used for CFD Analysis

Fig. 10. Pressure Contours for Muffler Model 1

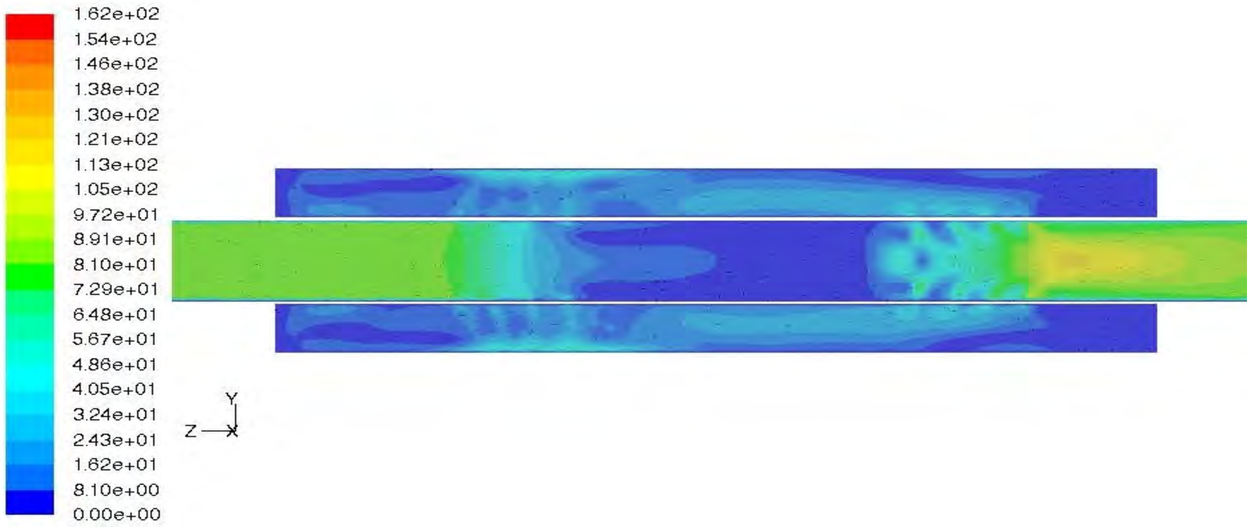

Fig. 11. Velocity Contours for Muffler Model 1

The commercial muffler model 1 shown in Figure 9 is having same porosity on both the sides of baffles. The pressure contour diagram is shown Figure 10. It shows the pressure distribution across the muffler. The pressure is high on the inlet side whereas it is low on the outlet side. As the number of holes is less in this muffler the backpressure provided by the muffler is very high. Figure 11 shows the velocity of air through the muffler.

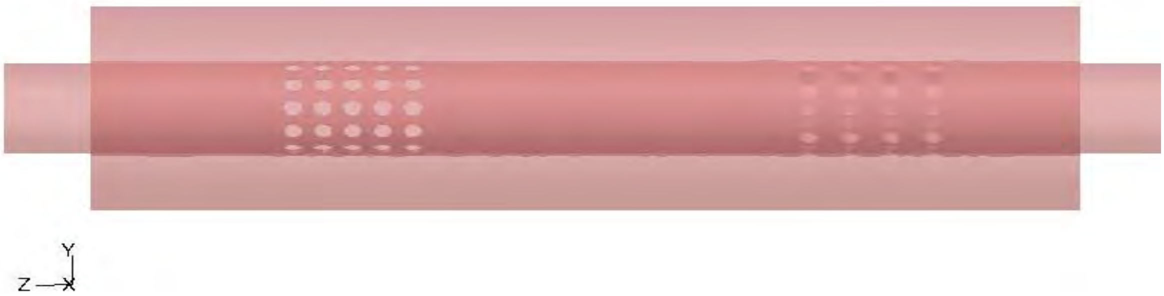

Fig. 12. Muffler CAD Model 2 for CFD analysis

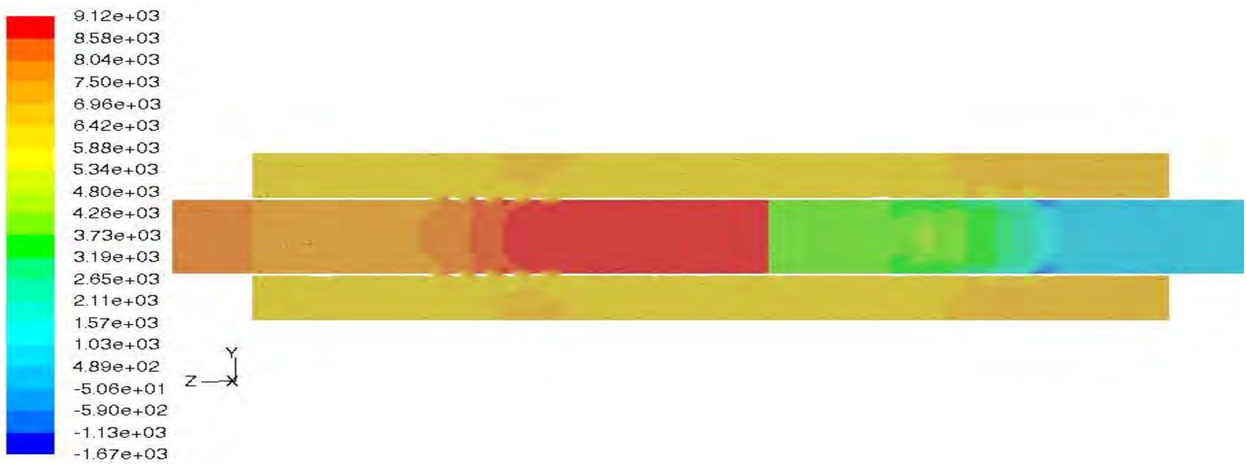

Fig. 13. Pressure contours for Muffler Model 2

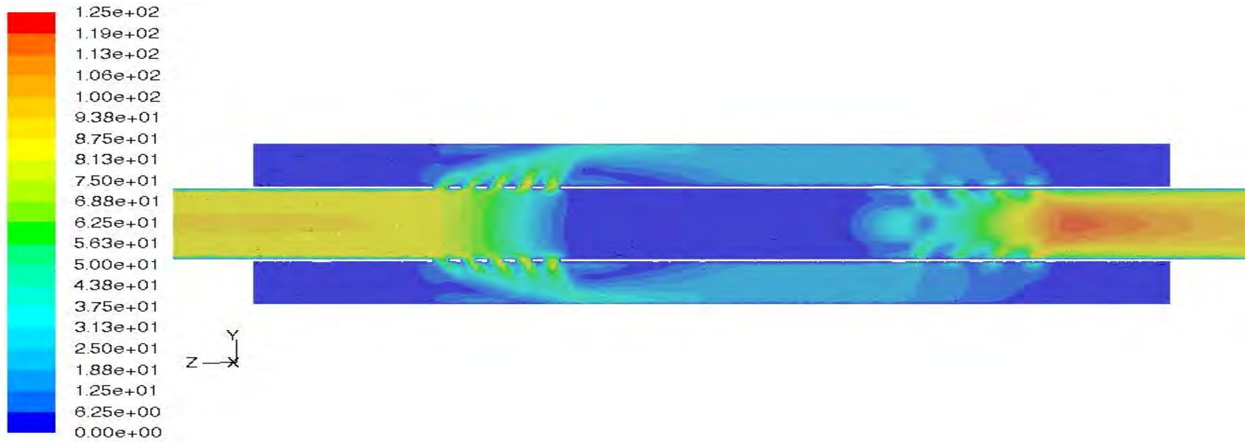

Fig. 14. Velocity Contours for Muffler Model 2

The commercial muffler model 2 shown in figure 12 is having different porosity on both the sides of baffles. The porosity on inlet side is same as that of Model 1 whereas it is less on outlet side. The pressure contour diagram is shown Figure 13. It shows the pressure distribution across the muffler. Comparing the Pressure contour diagrams for Model 1 and Model 2, it can be seen that the pressure that is built on outlet side of Model 2 is higher than that in Muffler 1. So, the backpressure is lesser in Model 2. Figure 14 shows the velocity of air through the muffler. So, in case of Model 2 it can be seen that since the number of holes on outlet side is higher the air is moving with high velocity.

Fig. 15. Muffler CAD model 3 for CFD analysis

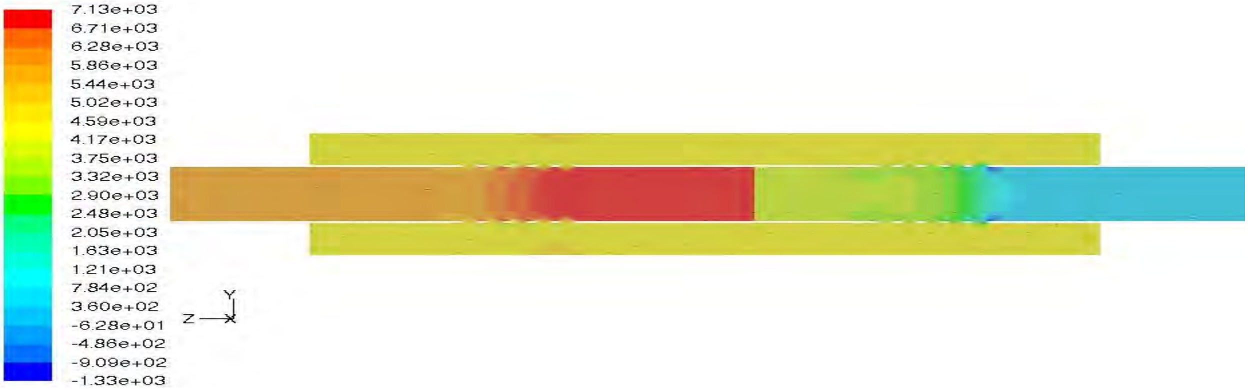

Fig. 16. Pressure Contours for Muffler Model 3

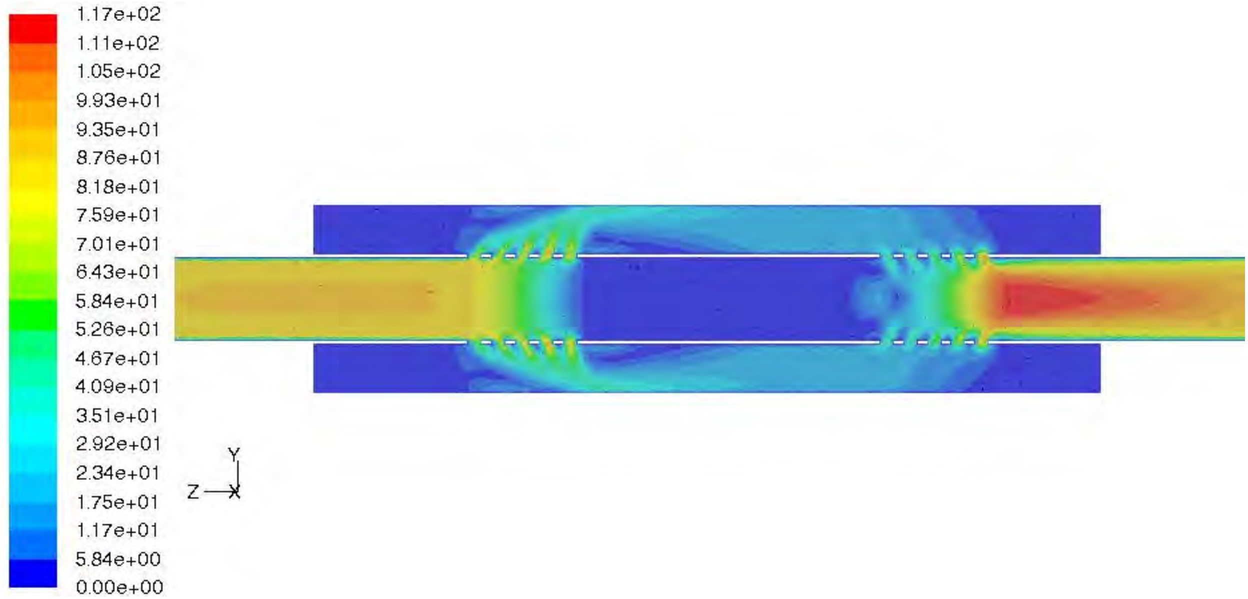

Fig. 17. Velocity Contours for Muffler model 3

Figure 15 shows the commercial muffler model 3 for CFD analysis. As seen from the figure 15 the porosity is higher in muffler model 3 as compared to Muffler Model 1 and 2. The Pressure and velocity contour diagrams are shown in Figure 16 and Figure 17 respectively. There is noticeable effect in the pressure distribution and velocity distribution. The velocity is quite high in Muffler model 3.

The mufflers that have been used for experimental work have a change in porosity. So it can be seen from the pressure contours diagrams of each muffler the pressure that is being built up due to the components of the muffler is higher when the porosity is smaller as in case of Muffler model 1 and backpressure is smaller when porosity is increased.

Also from velocity contour diagrams it can be concluded that, for first case(Model-1), the velocity of the air moving through the muffler is small so the pressure built up is high, whereas in second(Model-2) and third(Model-3) case the velocity is higher throughout the muffler so the pressure built up is lower.

V. CONCLUSIONS

The various dimensions of the muffler are varied keeping some dimensions constant and then the effect on Backpressure is observed. It can be seen that the backpressure varies nonlinearly and it cannot be predicted by any equation.

Varying the porosity of the muffler has pronounced effect on the backpressure. The Backpressure is reduced greatly if the porosity is doubled. Also if the diameter of the hole is increased the backpressure decreases sharply. The change in diameter of holes has remarkable effect. There is sharp change in backpressure values even if hole diameter is slightly changed.

Three mufflers were tested in lab with varying porosity. There was a sudden decrease in backpressure values as porosity was increased. The CFD values and Experimental values are in good agreement with each other.

ACKNOWLEDGEMENTS

The technical support and Experimental facility from Automotive Research Association of India (ARAI), Pune is gratefully acknowledged. The first author would like to thank Prof. Amol B Ubale from Symbiosis Institute of Technology, Pune for his useful suggestions during the project and also Mr.Vikram A Tandon from ARAI, Pune for his help in ANSYS Fluent software.

REFERENCES

[1] Key Fonseca De Lima, Arcanjo Lenzi, Renato Barbieri; “The Study of Reactive Silencers by Shape and Parametric Optimization Techniques”, Applied Acoustics 72 (2011) pp.142–150.
[2] Jesus Blanco, Andrew Earnshaw; “Study Of the Noise Characteristics of Motorcycle Silencers”, SAE 2001 World Congress Detroit, Michigan March 5-8, 2001.
[3] John Perkins, Jay Robichaud, Brian Ellis, “Advanced Two-Stroke Tuned Exhaust System” Mee 487 & 488 Design Iii &Iv”.
[4] Peter Hield; “The Effect of Backpressure on the operation of Diesel Engines”, Maritime Platform Division, Defense Science and Technology organization, DSTO-TR-2531, Australian Government (Unclassified).
[5] Ying-Li Shao; “A Study on Exhaust Muffler Using a Mixture of Counter phase-Counteract And Split-Gas Rushing”, Advanced In Control Engineering And Information Science, Engineering 00 (2011) pp.4409-4413.

[6] S.N. Panigrahi, M.L. Munjal; “A Generalized Scheme for Analysis of Multifarious Commercially Used Mufflers”, Applied Acoustics 68 (2007) pp.660–681.

[7] Omid Z. Mehdizadeh, Marius Paraschivoiu; “A Three-Dimensional Finite Element Approach for Predicting the Transmission Loss in Mufflers And Silencers With No Mean Flow”, Applied Acoustics 66 (2005) pp.902–918. International Journal Of Engineering And Innovative Technology (IJEIT) Volume 2, Issue 1, July 2011.

[8] Ansys Users Guide; “ http://hpclab.iitgn.ac.in/doc/wb2_help.pdf”

[9] Ansys Theory Guide; “http://cdlab2.fluid.tuwien.ac.at/LEHRE/TURB/Fluent.Inc/v140/flu_th.pdf ”